Simulation Comparison Between PTC/Creo and Professional CAE Software

Li-xin WANG*, Jun-hui LIU and Lin-kai CHEN

School of Mechanical Engineering, Zhengzhou University, Henan Zhengzhou 450001, China

*Corresponding author

Keywords: PTC/Creo, Mechanism analysis, Structure analysis, Adams, Ansys.

Abstract. CAE is an indispensable tool for mechanical design engineers, and is widely used in product development process. This paper discuss the differences and linkages of the two most commonly used mechanism simulation and strength calculation in the design process. For the same model respectively, simulations using PTC/Creo Vs. Ansys and PTC/Creo Vs. MSC Adams are conducted. The results show that PTC/Creo software is simple to use, easy to modify and accurate enough upon the simulation. However, it is a bit rougher and shallower than the professional CAE software. So, for mechanical design engineers, PTC/Creo simulation tools should be preferred, and the profound analysis should be the duty of professional CAE engineers.

Introduction

With the development and popularization of CAD technology, three-dimensional software design is more and more commonly used in modern mechanical design. At present, the general understanding is that CAD software is responsible for three-dimensional modeling of parts, components and machine assembly, and static interference analysis. Although CAD software also provides integrated CAE tools (mechanism simulation, structural analysis, thermal analysis, etc.). However, most people are accustomed to the simplified model, and export the model to the professional finite element analysis software (Ansys, Nastran, Abaqus) and virtual prototyping software (MSC.Adams, LMS Virtual.Lab, etc.) for analysis. If some problems are found, the process will return to the CAD Package.

Nowadays, the development of CAD software is very fast, Does its integrated CAE function meet most of the mechanical design needs? To get the answer, we have carried out some exploration.

CAE Analysis Ability Comparison

Table 1. Comparison of simulation of the mechanism.

<table>
<thead>
<tr>
<th>Function</th>
<th>PTC/Creo</th>
<th>Adams</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modeling</td>
<td>Accurate</td>
<td>reduced</td>
</tr>
<tr>
<td>Flexible body modeling</td>
<td>complex</td>
<td>simple</td>
</tr>
<tr>
<td>Model modification</td>
<td>simple</td>
<td>complex</td>
</tr>
<tr>
<td>Computing power</td>
<td>common</td>
<td>powerful</td>
</tr>
<tr>
<td>Analysis steps</td>
<td>simple</td>
<td>complex</td>
</tr>
</tbody>
</table>

Table 2. Comparison of structure analysis.

<table>
<thead>
<tr>
<th>Function</th>
<th>PTC/Creo</th>
<th>Ansys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modeling</td>
<td>convenient</td>
<td>complex</td>
</tr>
<tr>
<td>Model modification</td>
<td>simple</td>
<td>complex</td>
</tr>
<tr>
<td>Grid type</td>
<td>few</td>
<td>many</td>
</tr>
<tr>
<td>Meshing</td>
<td>sketchy</td>
<td>detailed</td>
</tr>
<tr>
<td>Calculating time</td>
<td>short</td>
<td>long</td>
</tr>
<tr>
<td>Results</td>
<td>common</td>
<td>detailed</td>
</tr>
</tbody>
</table>

As we all know, there are many kinds of CAE software, and also there are many differences in the
analysis methods and optimization. In this paper, we mainly study PTC/Creo, MSC.ADAMS and ANSYS. Therefore, some differences\textsuperscript{[1-7]} can be acquired by the help file in the official website of Creo, Adams, ANSYS, as well as the relevant BBS and the corresponding reference. The differences are shown in the table 1 and table 2.

**Comparison of Simulation Analysis**

For mechanical design engineers, in the design process, the two most common occasions are the various characteristics of the simulation of the mechanism, as well as the strength, stiffness and stability of important parts and structures. So, the paper makes the analysis and comparison of the use of CAD simulation module and professional CAE software. The following two aspects have been done: the stress and deformation of the parts, the kinematics and dynamics of the mechanism.

**Structural Analysis of a Gear**

Gear is widely used in industrial production and mechanical operation, so this paper takes the finite element analysis of cylindrical gear as an example.

**Stress and Deformation Analysis of a Gear by Creo.** Gear is the most prone to failure in the gear reducer, thus in the gear reducer for finite element analysis, the finite element analysis of the gear is particularly important.

In the actual working conditions, we start to calculate the gear meshing force. When the involute gear is engaged, it can generally be simplified for line contact. So, when the gears are engaged, the circumferential force and the radial force which are applied to a single tooth are equivalent to the resulting line load. Because the constraints of the gear are fixed by the key, the three sides of the keyway are fully constrained. Based on the above analysis, the "Simulate" function in the "application" module of Creo is used to define the material, constraints and loads. Using the AutoGEM function in the fine model module, the grid size is defined and automatically meshed. Next, we use the "analysis and research" module, to create new static analysis options, run the analysis check, detect whether the error exists, and get the maximum stress. The stress cloud is shown in the Figure 1.

According to the stress contour in the Figure 1.

The maximum stress of the model is 2.966MPa.
The minimum stress of the model is 2.005E-05MPa.
It is also known that the displacement contour of the model is shown in the Figure 2.
So we can get the maximum displacement is 3.4206E-06mm from Figure 2.

**Finite Element Analysis of Gear by ANSYS.** The model of the gear is saved as IGES format, and imported into ANSYS software by the IMPORT function of ANSYS\textsuperscript{[9-10]}. Then, the model is defined the same material properties, cell type, and mesh size\textsuperscript{[12]} as Creo, and the units are N, m.

Load and constraint types for gears, and the size and application of the values are consistent with
Creo. Creating a new static analysis, the results are shown in Figure 3 and Figure 4.

![Figure 3. Gear stress contour by ANSYS.](image1)

![Figure 4. The displacement contour of gear by ANSYS.](image2)

By comparing the analysis results of the two software in Table 3, the finite element analysis contour is basically the same, the results of the analysis is little difference in the range of error.

<table>
<thead>
<tr>
<th>Data Type</th>
<th>Creo</th>
<th>ANSYS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum stress</td>
<td>2.966MPa</td>
<td>3.84577MPa</td>
</tr>
<tr>
<td>minimum stress</td>
<td>2.005E-05MPa</td>
<td>3.93E-05MPa</td>
</tr>
<tr>
<td>maximum displacement</td>
<td>3.421E-09m</td>
<td>2.37E-09m</td>
</tr>
</tbody>
</table>

**Simulation of Aour-Bar Mechanism**

In order to reflect the advantages of CAE integration analysis, it is theoretically possible to carry out the simulation of single stage cylindrical gear reducer in Creo, and obtain a series of dynamic data, such as speed, acceleration and so on, and compare the data with the kinetic data that is introduced into Adams to illustrate the practicality of Creo's mechanism simulation. However, for single-stage gear reducers, the kinetic data is too simple, and there are specialized Machinery modules in ADAMS that can quickly establish gears, belts, chains and other mechanisms. Moreover, there are four types of modeling of cylindrical gears in Adams, namely, coupling, simplification, detail, 3D contact, and almost all types of gear simulation. But the gear connection in Creo method is relatively small. Therefore, in order to reflect the simulation ability of Creo, the common four-bar mechanism is analyzed, and the comparison of the results of the two simulation software is obtained.

**Creo Simulation of Four - Bar Mechanism.** In PTC Creo, the common four-bar mechanism is built by the existing size and assembled. The connection of the four-bar mechanism is defined by the "mechanism" function in the "application" module in Creo, and the quality, density and gravity direction of the member are also defined[11]. According to the result analysis module, a bar is selected, and the displacement, velocity and acceleration motion curve and the measured value are obtained. The following is the example of connecting rod D.

After completing the above definition, the next steps is that clicking "Mechanism Analysis", selecting kinematics in the analysis type, creating a new kinematics analysis, selecting the termination time is 5, the minimum interval of 0.01 and running. After all is done, we can observe the movement of four-bar mechanism.

Motion analysis results

1) Position

Create new measurement, select the location of the center of the D rod as the research object, select the measure1 and AnalysisDefinition1, and click on the top of the drawing results. Then, the result is shown in Figure 5.
2) Velocity
Create new measurement, select the velocity of the center of the D rod as the research object, and select the measure2 and AnalysisDefinition1, drawing the results of graphics, as shown in Figure 6.

3) Acceleration
New measurement, select the acceleration of the center of mass of the D rod as the research object, and select the measure3 and AnalysisDefinition1, draw the figure as shown in Figure 7.

**ADAMS Simulation of Four - Bar Mechanism.** The four-bar mechanism model in Creo is saved as Parasolid (*.x_t) format, and renamed to 'siganjigou', then the model is imported into Adams by the Adams interface\(^8\), and created a model named signanjigou. Then the model is materialized, and defined the gravity direction and the density of the body. The kinematic connection between the bodys is defined as revolute, the drive is attached to the rotation of the
shortest rod, and the numerical value is matched to Creo. The model is validated and simulated, According to the simulation results, the position, velocity and acceleration curves of the center of the D rod bar are obtained, and there are respectively shown in Figure 8 Figure 9 Figure 10.

![Figure 8. Displacement curve of D by ADAMS.](image1)

![Figure 9. Velocity curve of D by ADAMS.](image2)

![Figure 10. Acceleration curve of D by ADAMS.](image3)

By comparing the kinematic curves of Creo and Adams, both of them are consistent and highly consistent, which shows the consistency of the two software analysis results.

**Conclusion**

In most cases, modern CAD software sets modeling, simulation and analysis as a whole, and once the design intent needs to be modified or an error occurs, CAD integration analysis and modification is particularly simple and convenient. And in the analysis, it is not necessary for an engineer to have too much knowledge of finite element and multi-body dynamics. he can rapidly analyze the characteristics of the mechanical system and the components of the system which engineers often pay close attention to.

Only in some complex or special circumstances, the results require relatively accurate, CAD integrated software analysis capabilities may no longer meet the requirements, which requires the use of professional CAE analysis to solve.

**References**


[6] PTC Creo. PTC Creo3.0 Helpcenter[DK], 2014